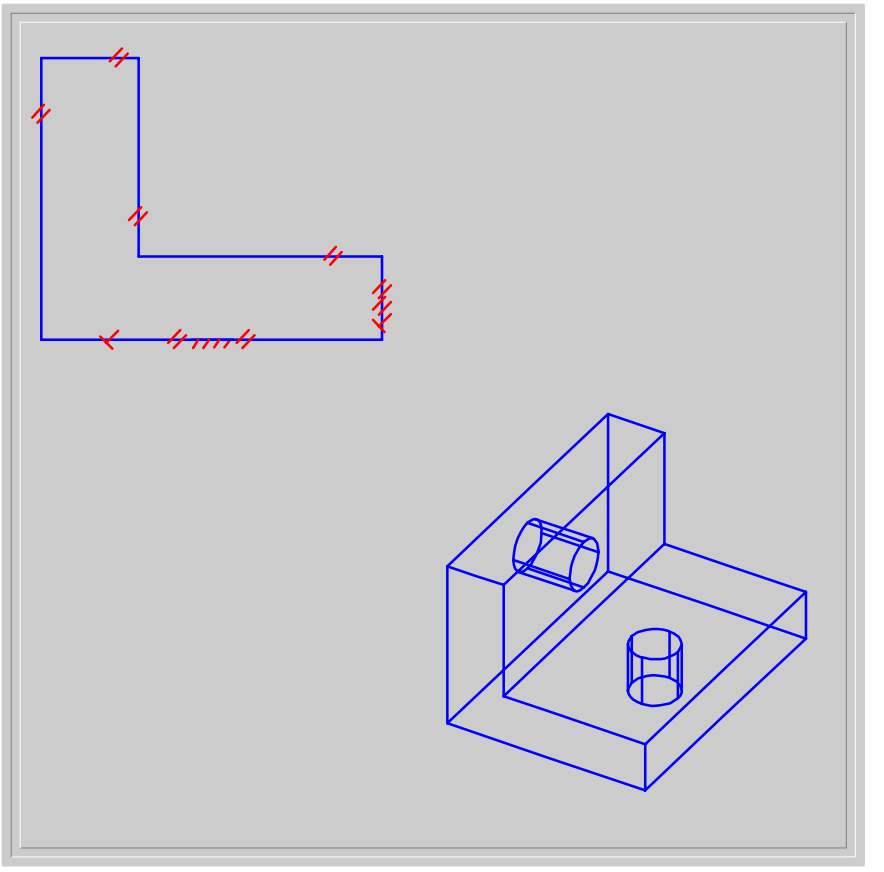


Creating Parts

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- create a simple part
- add a cut-out feature
- view a part's history
- modify a part
- name a part
- put a part away to a bin

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

  Introducing the I–DEAS Interface
  –and–
  Quick Tips to Using I-DEAS

Setting your defaults

What:

Before continuing, set the following default options for this tutorial.

How:

Options
Preferences...

Preferences form


Modeler/Assembly...

Modeler/Assembly Preferences form


Drag for extrude creation (off)


OK


OK

Why:

With these recommended settings, the tutorial steps will work as documented. Other settings may cause minor changes in the required steps.


For more information, use *Help, on Context* and then pick the specific item of interest.

If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.

File
Open

Open Model File form

Model File name: any unique name
OK

Make sure you're in the following application and task:

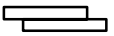
Design, *Manufacturing*, or *Simulation*
Modeler

Set your units to mm.

Options
Units

mm (*milli newton*)

Save your model file.


File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:


No

Save only when the tutorial instructions tell you to––not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint
To reopen your model file to the previous save, press Control-z.

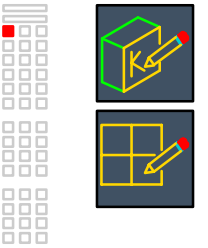
Creating a simple part is typically a 3-step process:

- picking a sketch plane
- sketching
- creating a feature

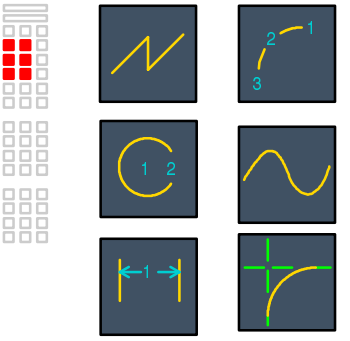
The Modeling task icons are organized to mirror the process of creating a part.

Some of the following icons will not be visible unless you click and hold the left mouse button to display the icon stack.

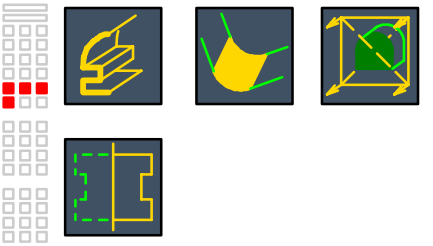
Picking a sketch plane



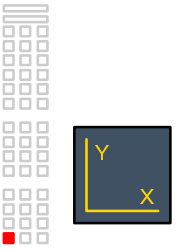
Sketching

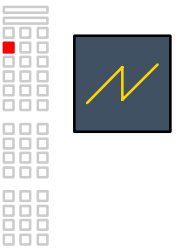


Creating a feature



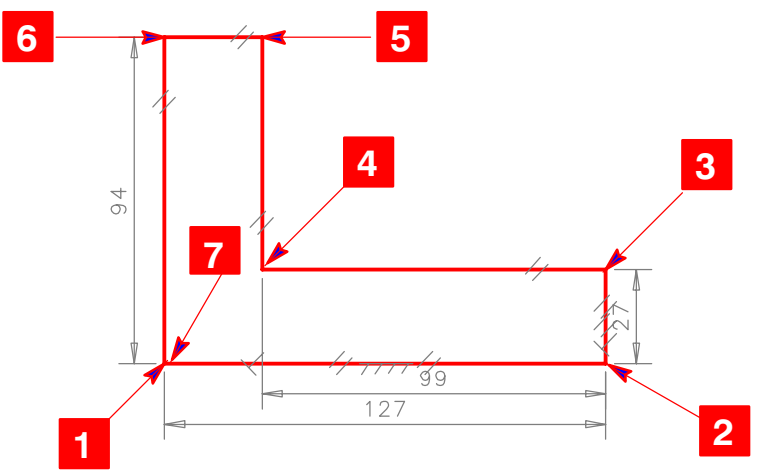
To create a simple part, you use the Modeling task icons. Begin by using the *Polylines* icon to sketch a closed shape. Don't worry about the dimensions that might appear on your sketch.



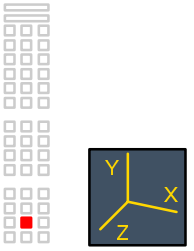


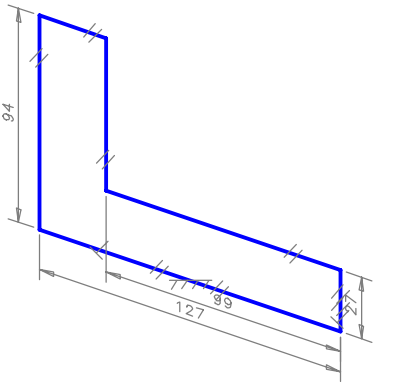
sketch the boundary as shown below, using the left mouse button to pick points 1 through 7

After picking point 7, the software assumes the boundary is complete. Terminate the command by pressing the middle mouse button.



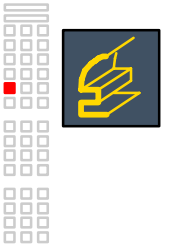
Change to isometric view.





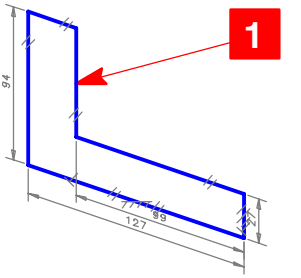
This will give you a better view of the next operation.

Next, extrude this shape a depth of 150mm.

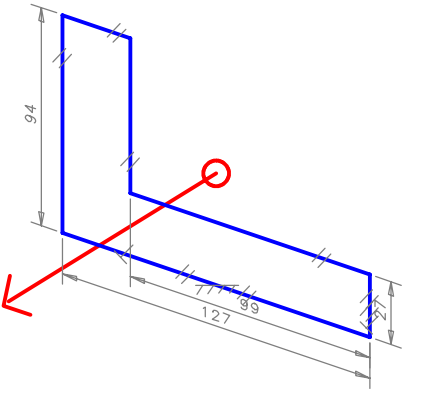


1 pick any line

(Done)



Move the Extrude Section form to get a better look at the sketch.

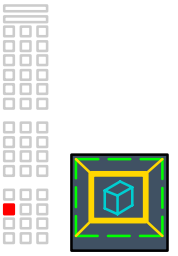


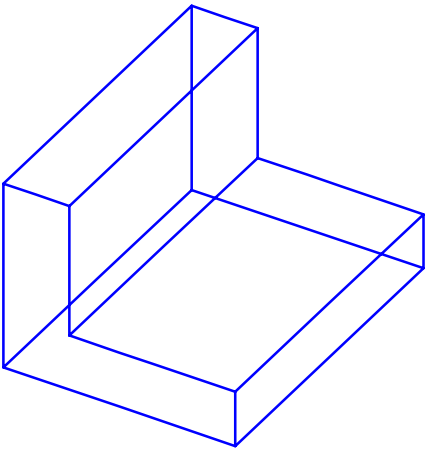
Extrude Section form

Distance: 150

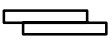
OK

Make sure the entire part fits on the screen.





Recovery Point



File
Save

Check *I-DEAS List*.
A message will tell you that your model file changes have been saved.

You just finished creating a simple part using the 3-step process of:

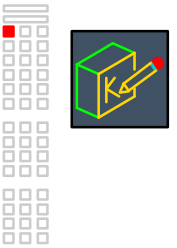
1. selecting (by default) the workplane
2. sketching
3. creating (by extruding) a feature

To add a feature to a part, you follow the same process, only you select the workplane differently. In this case, you will sketch on an existing part face.

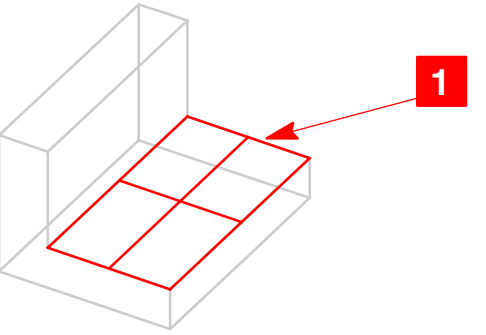
When you use the *Sketch in Place* icon, it will prompt you to pick a face. The face you pick will be outlined in the workplane color, and the original workplane will disappear. These are the graphical feedback clues that tell you that you are now working directly on the face of a part.

In this section, you'll add a cut-out feature to the part you just created by selecting a face and extruding a circle through the part.

First, use the *Sketch in Place* icon to pick a surface to sketch on.



1 pick anywhere on surface shown



Move the mouse around until the surface shown above highlights, then click the left mouse button.

Things to notice

If the following happens:

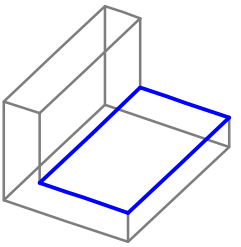
- the *Sketch on Face* icon is still highlighted
- the *I-DEAS Prompt* area is asking you to (Accept)
- the mouse pointer changes to a double crosshair

You should:

- press the middle mouse button, or press Return. Otherwise, when you go to the next step, you will not be sketching on the face.

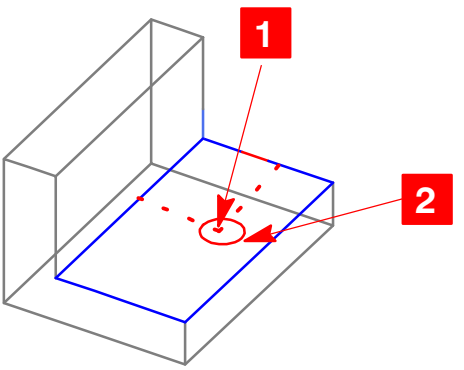
Next, sketch a circle on the surface selected. Don't worry about the size of the circle.

Don't sketch the circle if the face outline is not highlighted. This may mean you have not picked the surface to sketch on (see previous page).



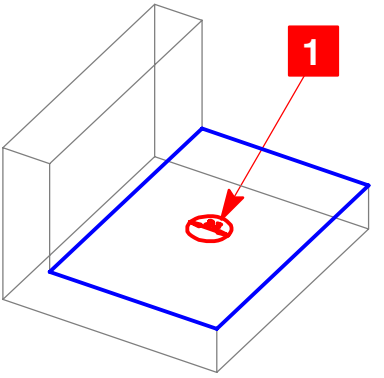
1 click where you want the center of the circle

2 click where you want the edge of the circle



(Done)

Next, cut the circle through the part.



1 pick circle

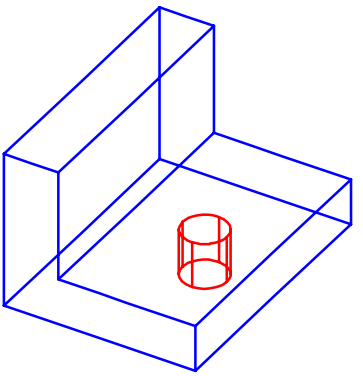
(Done)

Extrude Section form

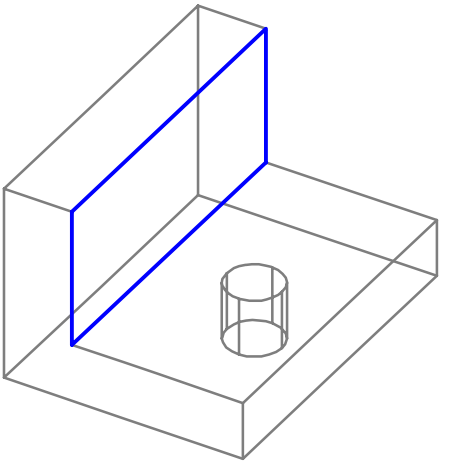
Cutout (toggle on)

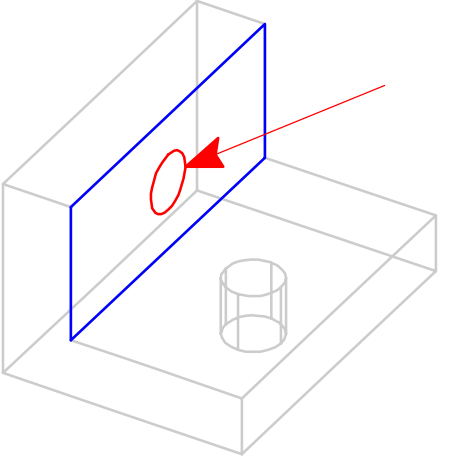
Depth:
Thru All

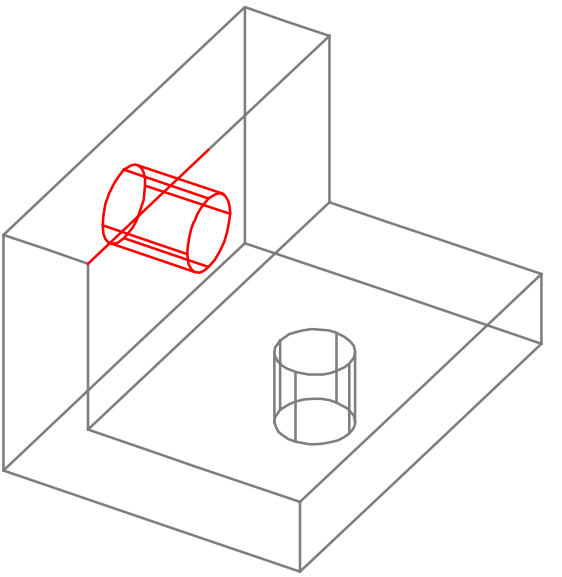
OK



Add a cut-out feature to the surface shown below. Follow the same steps used to create the previous cut-out.





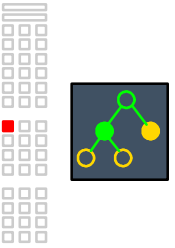


Recovery Point

File
Save

Every part you create with the Modeling task has a history – a record of the modeling events that were used to create it. You can view a history tree to see how a part was constructed.

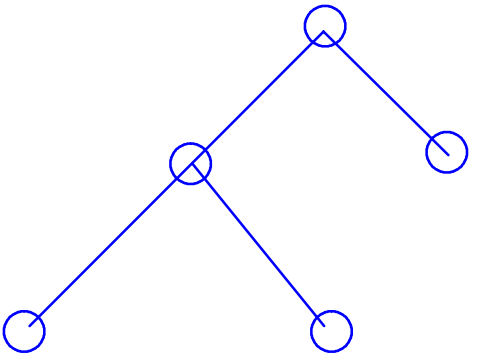
In later tutorials, you will learn how to use the history tree to select features to modify.



pick anywhere on part

(Accept)

History Access form

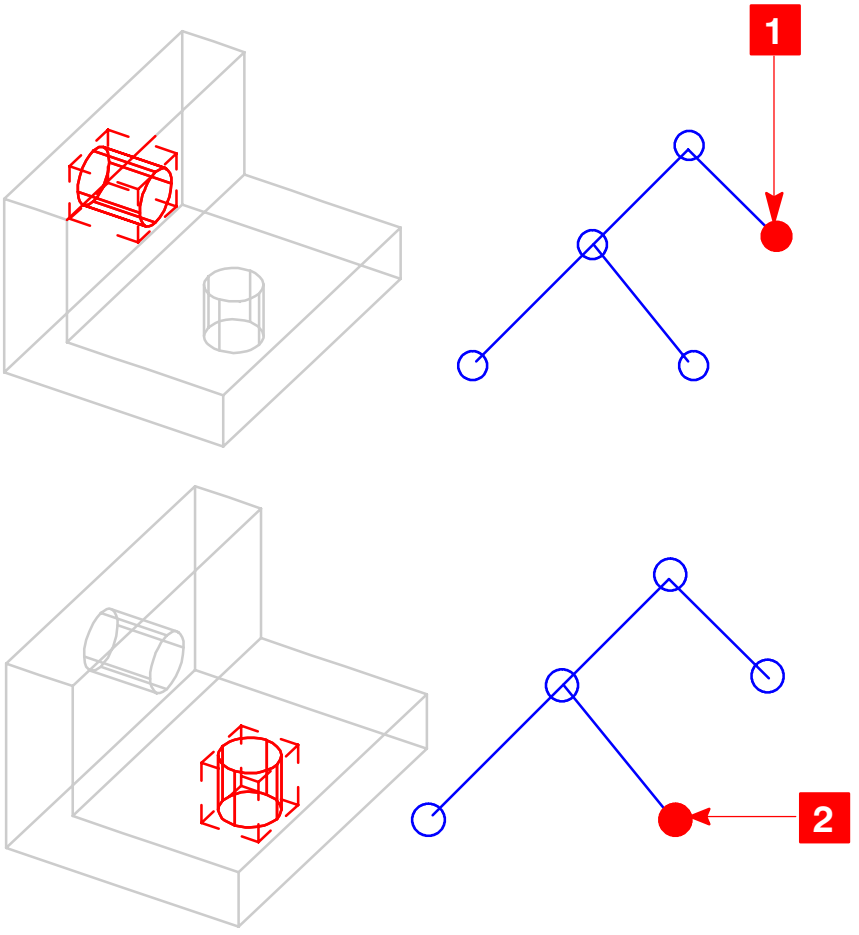


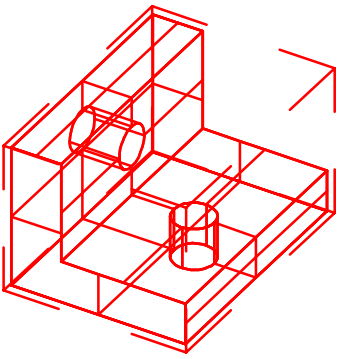
A history tree contains nodes, which typically include a parent and 2 children.

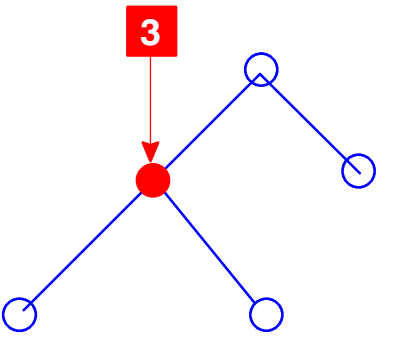
Do not dismiss the History Access form yet.

Click on the leaves and nodes on the History Access form displayed. See how the corresponding feature is highlighted on the part.

History Access form







History Access form

Dismiss

Deselect All

The primary use for the history tree is to modify features on the part. Using the history tree, you can select any feature for modification.

You'll be learning much more about the history tree in later tutorials.

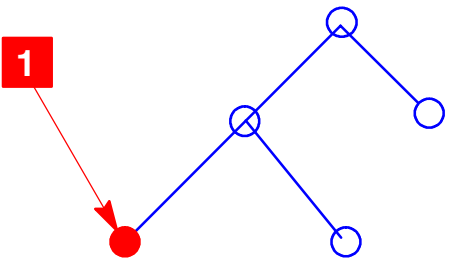
Access the History Access form.

 pick anywhere on part

 (Accept)

History Access form

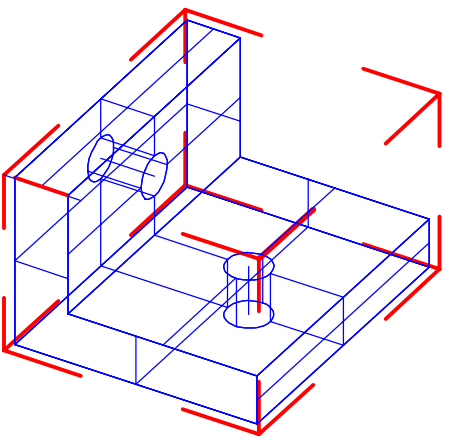
Select the first feature to modify.



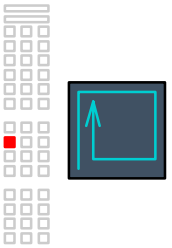
Dismiss the History Access form before continuing.

History Access form

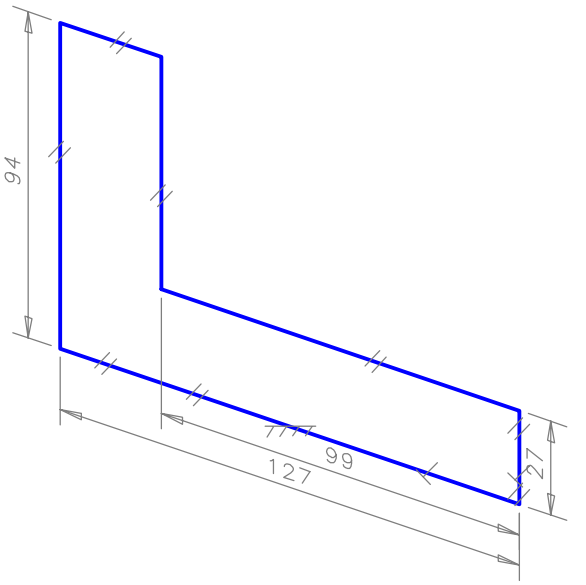
 *Dismiss*



Use the *Modify* icon to select the wireframe of that feature.



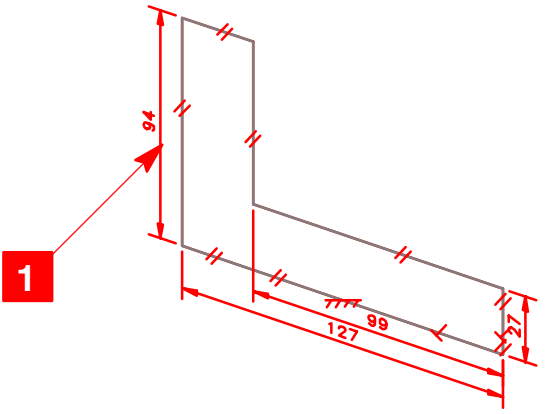
Wireframe



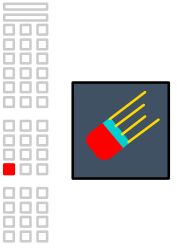
With the feature's original wireframe opened, you can now modify the dimensions or add new ones.

Delete all the dimensions (if there are any showing).

1 pick dimension

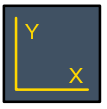


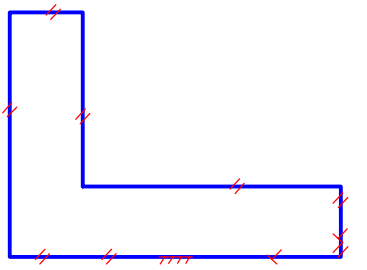

All



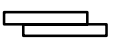

Yes

Switch to front view.





Recovery Point

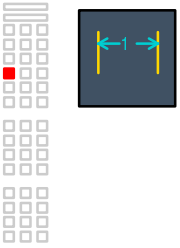

File
Save

Dimensions can be created:

- line –to– line (if lines are parallel)
- line –to– point
- point –to– point

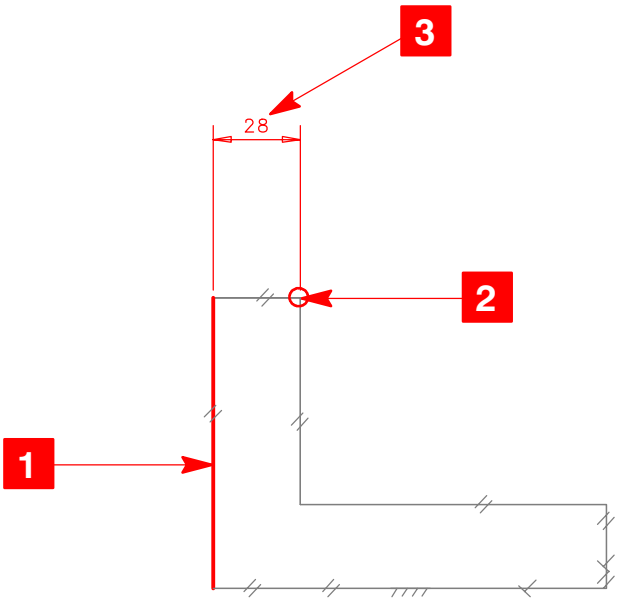
In many cases, line –to– point, which is the distance perpendicular from the line to the point, is preferred over point –to– point.

Practice creating dimensions as shown in each of the next few examples. Use line –to– point for each.



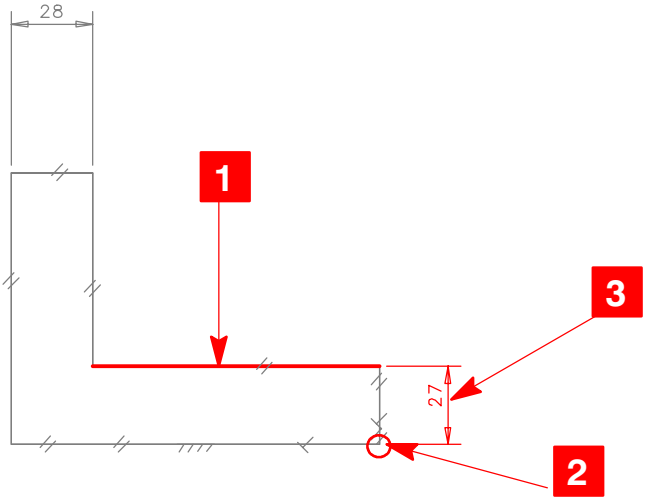
Example 1:

1. pick line
2. pick point
3. place the text



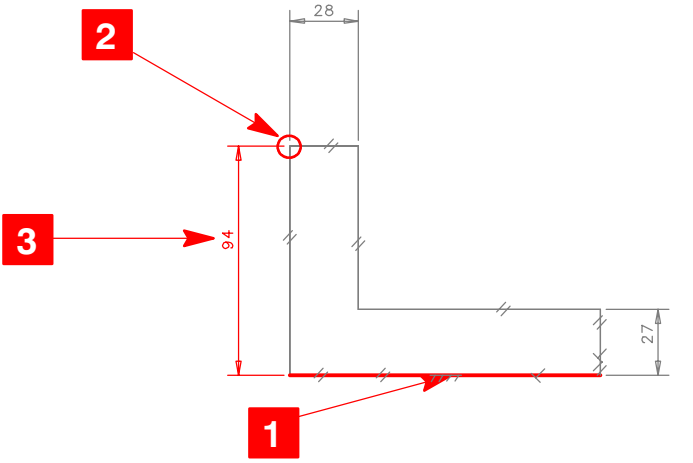
Example 2:

1. pick line
2. pick point
3. place the text



Example 3:

1. pick line
2. pick point
3. place the text



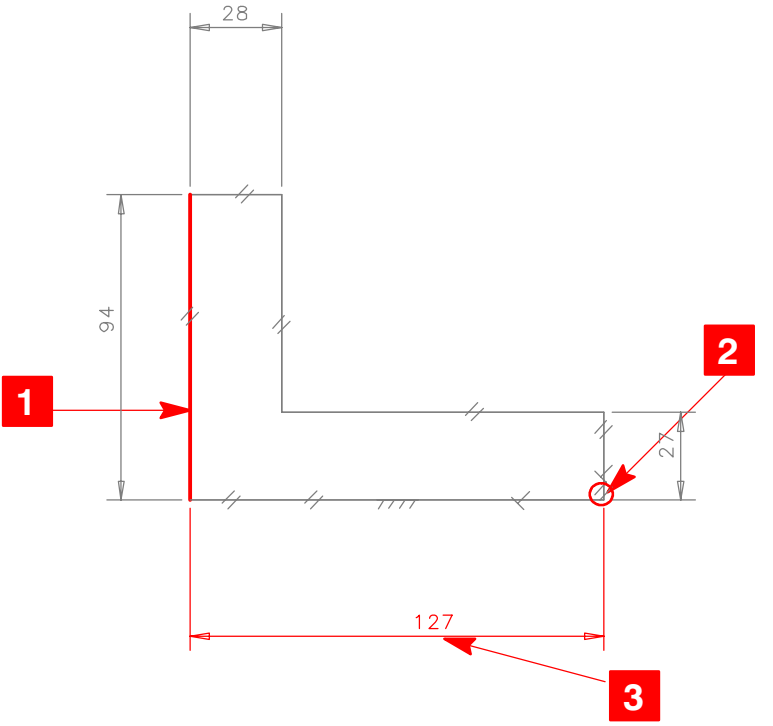
Example 4:

1 pick line

2 pick point

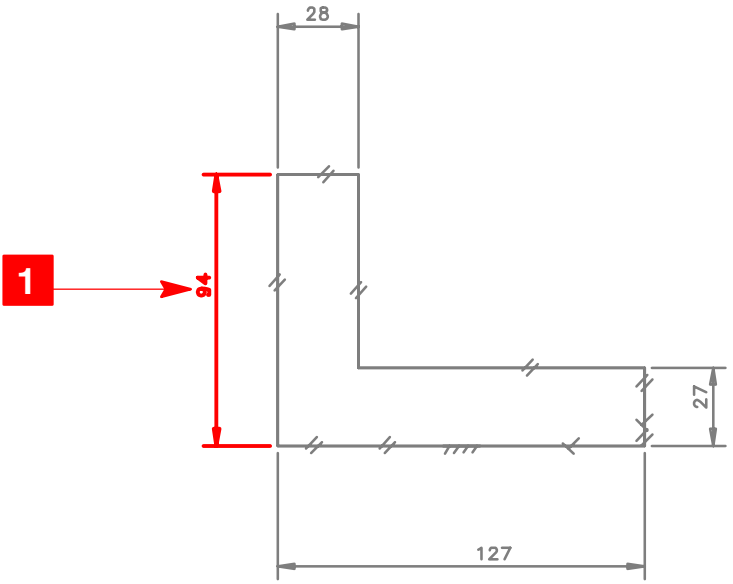
3 place the text

(Done)



Use the *Modify* icon to modify two of these dimensions.

1 pick dimension



Modify Dimension form

 = 100

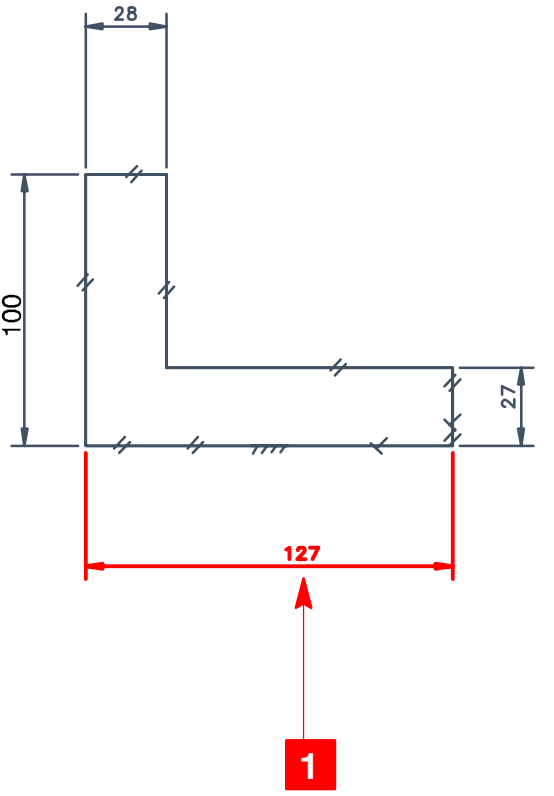
 *OK*

Things to notice

Don't worry if any of the lines go out of parallel when you modify a dimension. This is controlled by the parallel and perpendicular constraints that were created when you sketched the shape. Creating these constraints is covered in later tutorials.

Pick the second dimension to modify.

1 pick dimension



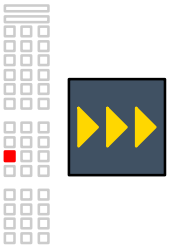
Modify Dimension form

= 125

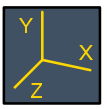
OK

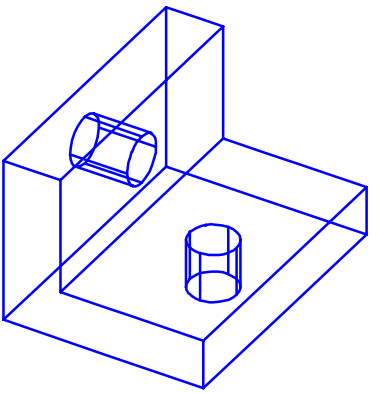
(Done)

To finish the part, update it twice. The first time updates the specific feature, the second time continues the update through the history tree.



Change to isometric view.





Recovery Point

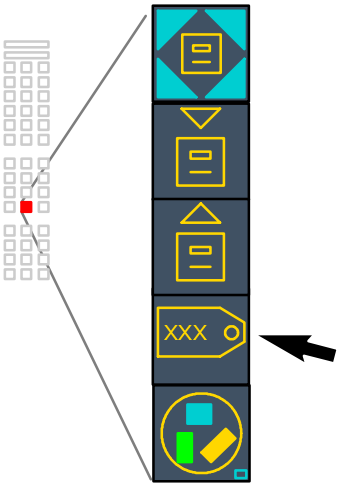
File
Save

More on this topic is covered in the tutorial “Modifying Parts.”

Name a part

Name the part Corner Bracket.

Pull down the icon stack and select the *Name Parts...* icon. Then release the mouse button.



pick anywhere on the part

Name form

Name: Corner Bracket

OK

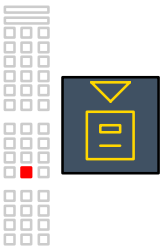
(Done)

Put a part away to a bin

Bins are storage containers similar to drawers in a desk. They're valuable tools that help you organize and manage your parts. When you open a new model file, there is only one bin, the default, or "main" bin. If you want more bins, you can add them to your work area.

Bins are covered in more detail in the tutorial "Managing Parts in Model Files."

Put the part away to the main bin.



pick anywhere on part

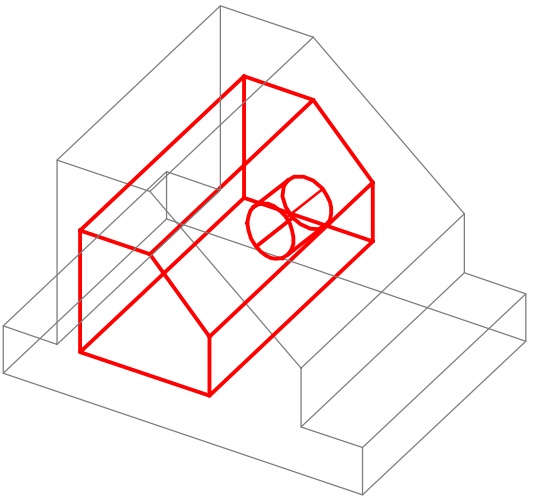
(Done)

Save your model file.

File
Save

Create a part similar to the one shown below. Add the two cut-out features to the surfaces shown. Refer to the 3-step process used in this workshop:

- pick a sketch plane
- sketch
- create a feature

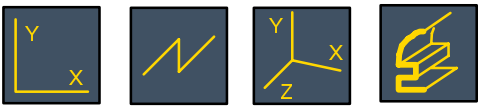


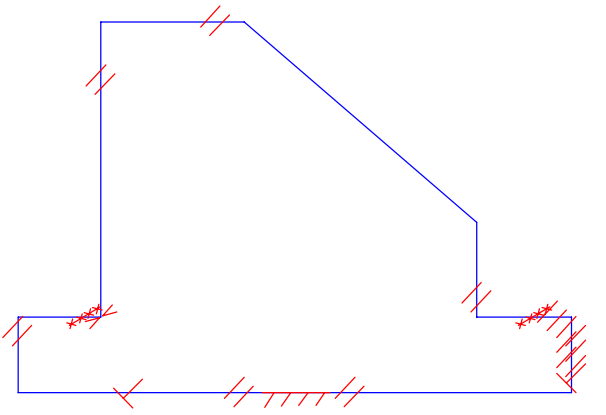
Try this on your own. You should be able to do this with the skills learned in this tutorial.

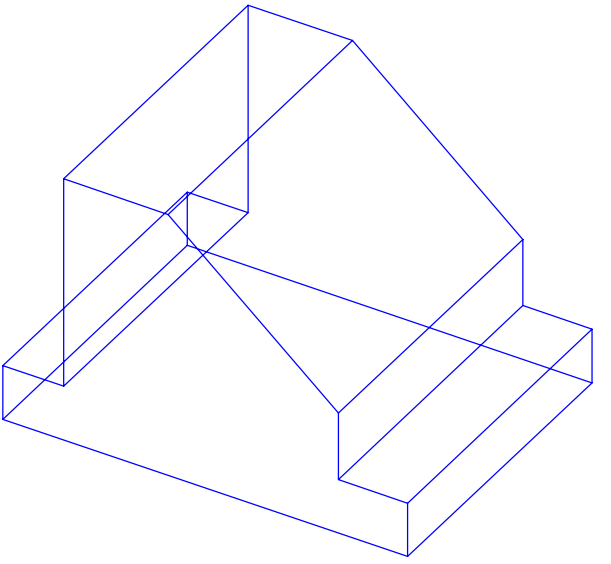
If you need help, refer to the next few pages, which give you hints on how to create the part shown.

If you would rather quit this tutorial now and try the "On your own" later, skip to the last page.

Step 1. Sketch the shape shown, extrude accepting all defaults, and switch to isometric view.

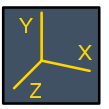


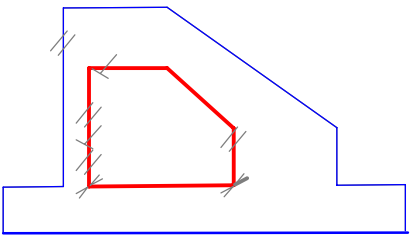


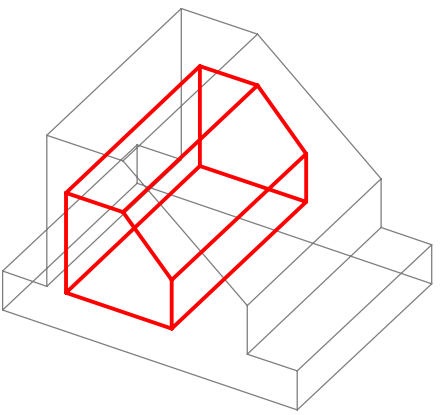


Step 2. Select *Sketch in Place*, switch to front view, sketch the polygon on the front face and extrude using *Cutout*, *Depth, Thru All*.







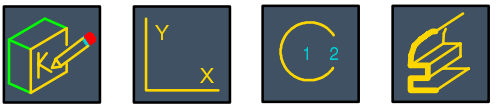


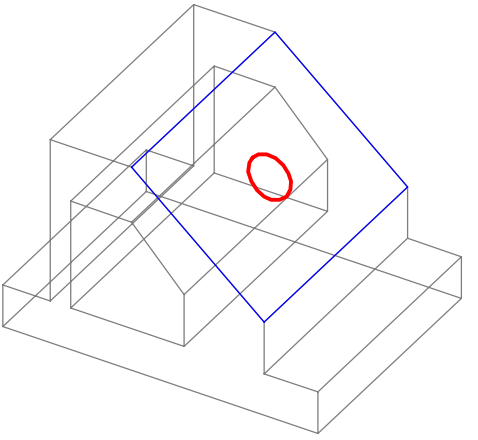
Another modeling technique would have been to add the inside wireframe in Step 1 and extrude both the shape and cutout at the same time.

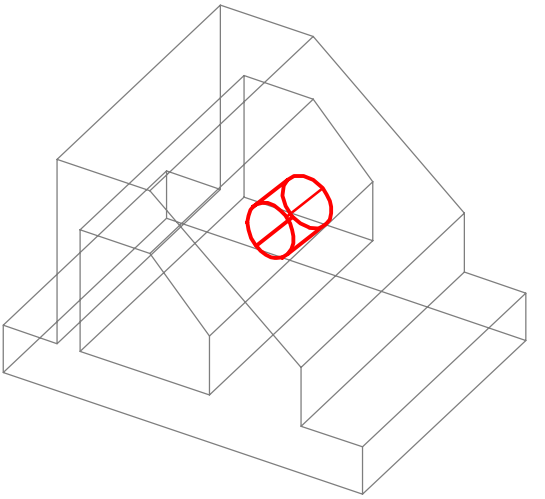
Step 3. Sketch the circle on the side face and extrude.

When you extrude the circle, use *Cutout*, but don't use *Depth, Thru All*. Look at the arrow and determine the appropriate length (*Depth*). Use the preview icon before you click *OK* on the Extrude Section form.

Use dynamic viewing (F3) to turn the part so you can get a better look at the arrow and the inside of the part.







Tutorial wrap-up

You have completed the Creating Parts tutorial.

Make sure you save your model file. The part you created is used in a later tutorial.

If you did the “On your own” exercise, you can delete or put away the part. This part is not used in any other tutorials.